

Finite element analysis (FEA) modeling on adhesive joint for composite fuselage model

ABSTRACT

In this paper, a finite element modeling via ABAQUS/Explicit simulation on a novel fabrication miniature composite fuselage structure is presented. The fuselage structure is modeled as a continuum composite layup that consisted of a woven C-glass fiber/epoxy 200 g/m² composite laminated [90 8] with the orthotropic elastic material properties and adhesively bonded butt joint. The adhesively bonded joint progression is modeled using cohesive elements technology. For the purpose of FEA modeling, an experiment of double cantilever beam (DCB) according to ASTM standard D5528 is performed to determine the adhesive mode-I critical toughness. The mode-I interlaminar fracture toughness data (G_I) are calculated and compared by four different methods according to the ASTM standard: BT, beam theory, MBT, modified beam theory, CC, compliance calibration method and MCC, modified compliance calibration method. The results indicate that ABAQUS/Explicit is able to reproduce satisfactory adhesive joint behavior using cohesive elements and collapse modes under crushing process.

Keyword: Adhesively bonded joint; Composite; Finite element analysis (FEA); Fracture toughness; Fuselage structure